Image Cover Sheet

CLASSIFICATION	SYSTEM NUMBER 498675
UNCLASSIFIED	

TITLE

AN EXAMPLE OF OBTAINING DETAILED STRESSES OF A PORTION OF A MAESTRO MODEL OF A SHIP STRUCTURE BY BOTTOM-UP AND TOP-DOWN ANALYSIS USING MAEST

System Number:

Patron Number:

Requester:

Notes:

DSIS Use only:

Deliver to: FF

National Defence / Défense nationale
Research and Development Branch / Bureau de recherche et développement

DREA CR/95/500

An Example of Obtaining Detailed Stresses of a Portion of a MAESTRO Model of a Ship Structure by Bottom-Up and Top-Down Analysis Using MAESTRO MG / DSA

by
D.R. Smith

D.R. Smith
Suite 707, 5959 Spring Garden Road
Halifax, Nova Scotia, Canada
B3H 1Y5

CONTRACTOR REPORT

Prepared for

Defence Research Establishment Atlantic

Centre de Recherches pour la Défense Atlantique

Canada

P498675.PDF [Page: 4 of 45]

THIS IS AN UNEDITED REPORT ON SCIENTIFIC OR TECHNICAL WORK CONTRACTED BY THE DEFENCE RESEARCH ESTABLISHMENT ATLANTIC OF THE RESEARCH AND DEVELOPMENT BRANCH OF THE DEPARTMENT OF NATIONAL DEFENCE, CANADA.

THE CONTENTS OF THE REPORT ARE THE RESPONSIBILITY OF THE CONTRACTOR, AND DO NOT NECESSARILY REFLECT THE OFFICIAL POLICIES OF THE DEPARTMENT OF NATIONAL DEFENCE.

PLEASE DIRECT ENQUIRIES TO:

THE CHIEF,
DEFENCE RESEARCH ESTABLISHMENT ATLANTIC,
P.O. BOX 1012,
DARTMOUTH, NOVA SCOTIA, CANADA
B2Y 3Z7

National Defence
Research and Development Branch

Défense nationale
Bureau de recherche et développement

DREA CR/95/500

An Example of Obtaining Detailed Stresses of a Portion of a MAESTRO Model of a Ship Structure by Bottom-Up and Top-Down Analysis Using MAESTRO MG / DSA

by
D.R. Smith

D.R. Smith
Suite 707, 5959 Spring Garden Road
Halifax, Nova Scotia, Canada
B3H 1Y5

Scientific Authority ____________ W7707-5-3292/01-HAL
Layton E. Gilroy — Contract Number

December 1995

CONTRACTOR REPORT

Prepared for

Defence Research Establishment Atlantic

Centre de Recherches pour la Défense Atlantique

Canada

Abstract

This report describes the method and results of a detailed analysis of a portion of a ship structure modelled by MAESTRO using bottom-up and top-down analyses. For the bottom-up method, a MAESTRO analysis of the structure was carried out to identify high stress regions. These regions were then modelled in detail using the program MAESTRO MG/DSA. The detail was created as a MAESTRO superelement. The superelement was automatically statically condensed and inserted into the original MAESTRO model equilibrium equations replacing the identified MAESTRO elements. A MAESTRO analysis of the modified model, using the same loading and boundary conditions as the original model, was run and detailed displacements and stresses were obtained using the superelement feature of the finite element program VAST. For the top-down analysis, the detail model created for the superelement was used as a stand-alone model. The displacements from the initial MAESTRO analysis were applied to the stand-alone model as boundary conditions and a finite element analysis using VAST was carried out. The results of the three analyses were compared. They showed that a much better definition of the local stress condition was obtained with the detail model than by MAESTRO alone, with little difference in the results between the bottom-up and the top-down methods.

Résumé

Ce rapport décrit la methode et les résultats d'une analyse détaillée d'une partie de la structure d'un navaire modélisée par MAESTRO en utilisant des analyses ascendante et descendante. Pour la methode ascendante, une analyse de la structue par MAESTRO a été réalisée. On a ainsi obtenu les déplacements et les efforts. La partie détaillée de la structure du navire a alors été modélisée par le programe MAESTRO MG/DSA en utilisant un super-élément de MAESTRO. Ce super-élément a été automatiquement condensé et intégré dans les équations d'équilibre du modéle original de MAESTRO en remplacant les éléments MAESTRO concernés. Une analyse par MAESTRO du modèle ainsi modifié dans des conditions identiques à l'analyse du modèle original a pu être réalisée. On a obtenu les déplacements et les efforts détaillés en utilisant le module super-élément du programme éléments finis VAST. Pour l'analyse descendante le modèle du détail de la structure créé pour le super-élément a été utilisé comme un modèle complètement isolé. Les déplacements fournis par la première analyse faite avec MAESTRO ont été appliqués à ce modèle en tant que conditions aux limites et une analyse par éléments finis a été conduite avec le code VAST. On a comparé les résultats des trois analyses. Ceux-ci ont montré que la définition des efforts locaux était bien meilleure avec le modèl du détail de la structure qu'avec MAESTRO tout seul, avec une petite différence au nieau des résultats entre les méthodes ascendante et descendante.

Contents

List of Figures

1 Introduction

This report describes both a bottom-up and a top-down finite element analysis of a structural detail in a portion of a ship structure which was modelled and analyzed using the computer code MAESTRO [1]. The stress results from the MAESTRO analysis were used to identify the high stress region that was to be investigated by refining the structural model locally. The detailed analyses were accomplished using the MAESTRO Graphics System [2]. MAESTRO Graphics is a graphics program which can operate in two modes. The MG mode is used to view MAESTRO models and results from a MAESTRO analysis. The DSA (Detail Stress Analysis) mode is used to perform a detailed stress analysis by first generating a refined model of the high stress region of a MAESTRO model. The refined model may be used as a superelement for a bottom-up analysis, or as a stand-alone model for a top-down analysis. In both methods of analysis, master nodes, which are shared or have identical coordinates as the nodes of the MAESTRO model, are identified graphically at the refined model boundaries . Additional nodes at the boundaries, created in the refinement process, are also identified and slaved to the dependent master nodes. In the case of the bottom-up method, the refined model is made a superelement by condensing out all the degrees of freedom and loads, except those at the boundaries. The condensed stiffness of the superelement then forms part of the MAESTRO equilibrium equation by-passing that of the MAESTRO elements it replaces. The modified MAESTRO model is then run to obtain displacements which are then used automatically by the finite element program VAST[3] to perform a superelement analysis.

For a top-down analysis, the displacements of the master nodes, from the initial MAESTRO analysis, are applied to the matching nodes of the stand-alone model as boundary conditions. Any loads that act on the refined model are also applied. A finite element analysis using VAST is automatically carried out. The stresses obtained with both methods provide greater precision than the initial MAESTRO analysis as a consequence of the more detailed grid in the refined model.

The procedure for generating and displaying the models are described in this report, along with the analytical results from the two methods, by recording the terminal session in the form of hard copies of the terminal screens.

2 Initial MAESTRO Analysis

An initial MAESTRO analysis of the MAESTRO model was conducted to obtain the global stresses and to identify the region that would benefit from a detailed stress analysis. The model file previously created using MAESTRO was EX1.DAT where EX1 was the chosen job name and DAT the extension assigned by the program. The analysis created the following files:

EX1.PLG contains the MAESTRO model and boundary conditions.
EX1.PLL contains the load data required for plotting loads.
EX1.PLD contains the model deformations.
EX1.PLS contains the element stresses.

2.1 Terminal Session to Display the Results from the MAESTRO Analysis

The terminal session was reproduced as a series of hardcopies of the windows used to generate the graphic displays desired. In each hardcopy of a window it is generally possible to see the buttons that have been activated.

(1) The initial window, as shown in Figure 1, places the program in the MG mode, and the user first selects or checks the file that holds the MAESTRO model.

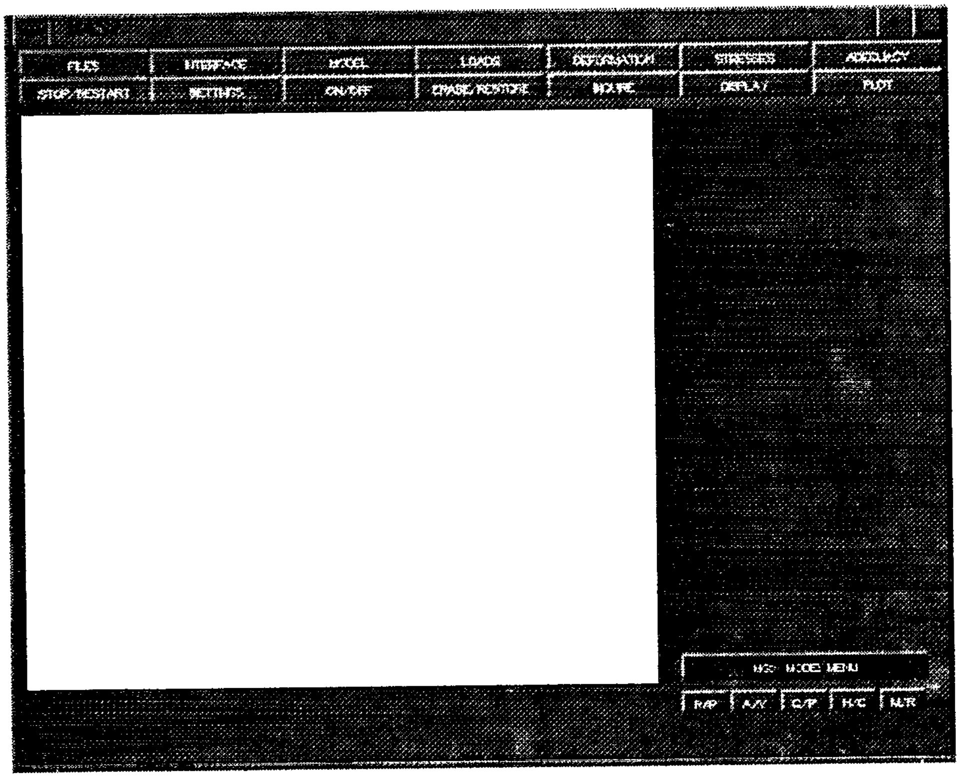

Figure 1: Initial Window of MAESTRO Graphics Program in MG Mode

(2) The file selection option was chosen as shown in Figure 2 by clicking on button FILE.

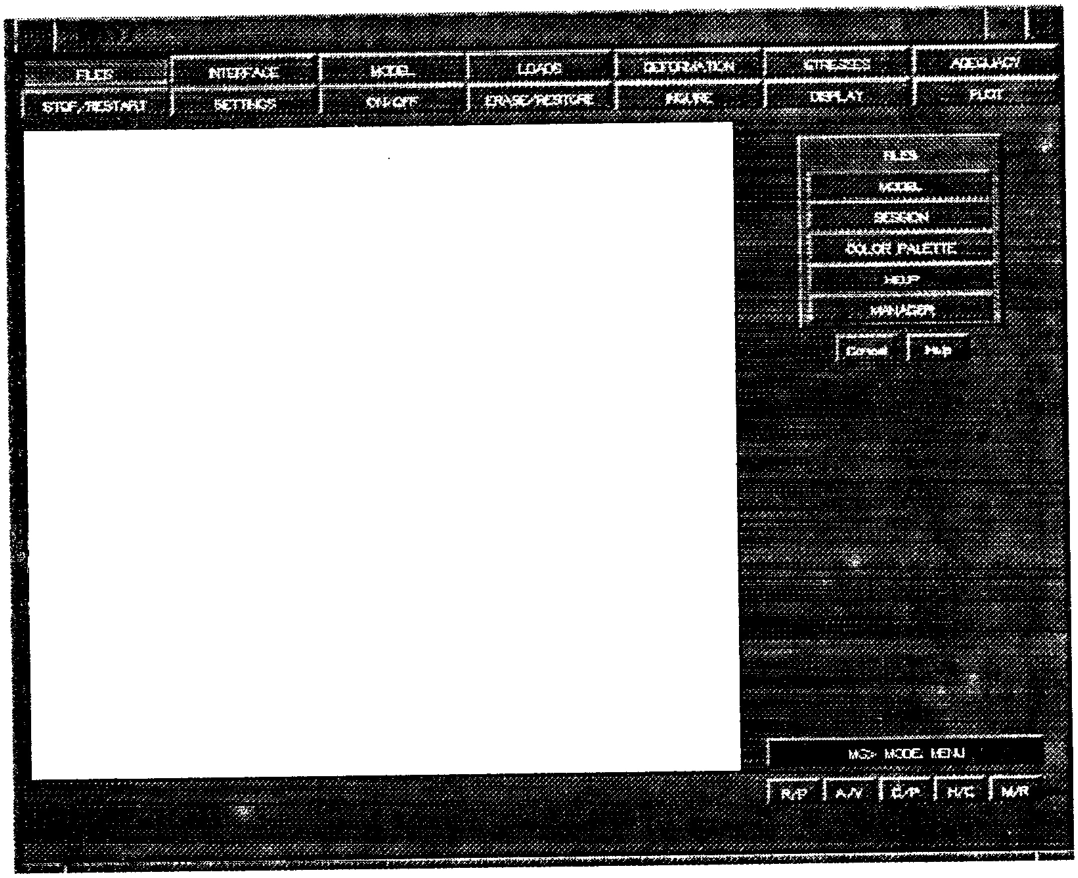

Figure 2: File Selection Option List

(3) Activating the MODEL button permitted the file name prefix for the MAESTRO files to be entered by clicking on MAESTRO Jobname. The job name prefix EX1 was entered from the keyboard, as shown in figure 3. This was followed by a click on OK to confirm the job name chosen.

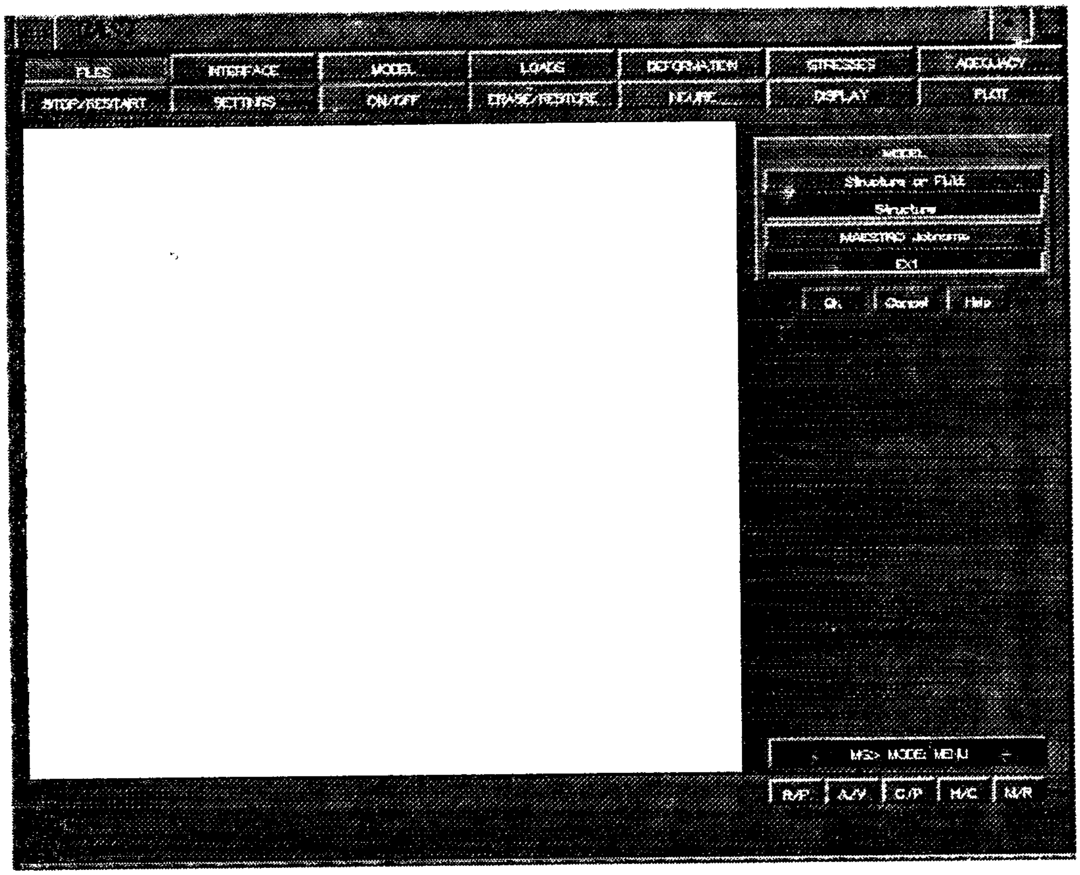

Figure 3: Confirmation of MAESTRO Graphics Program Job File Name

(4) After confirming the file, the next window listed the files PLC, PLE, and PLN at the bottom of the window. Their existence was confirmed by choosing 'Yes' with the cursor. This automatically produced the WIRE MESH plot of the model EX1 as shown in figure 4.

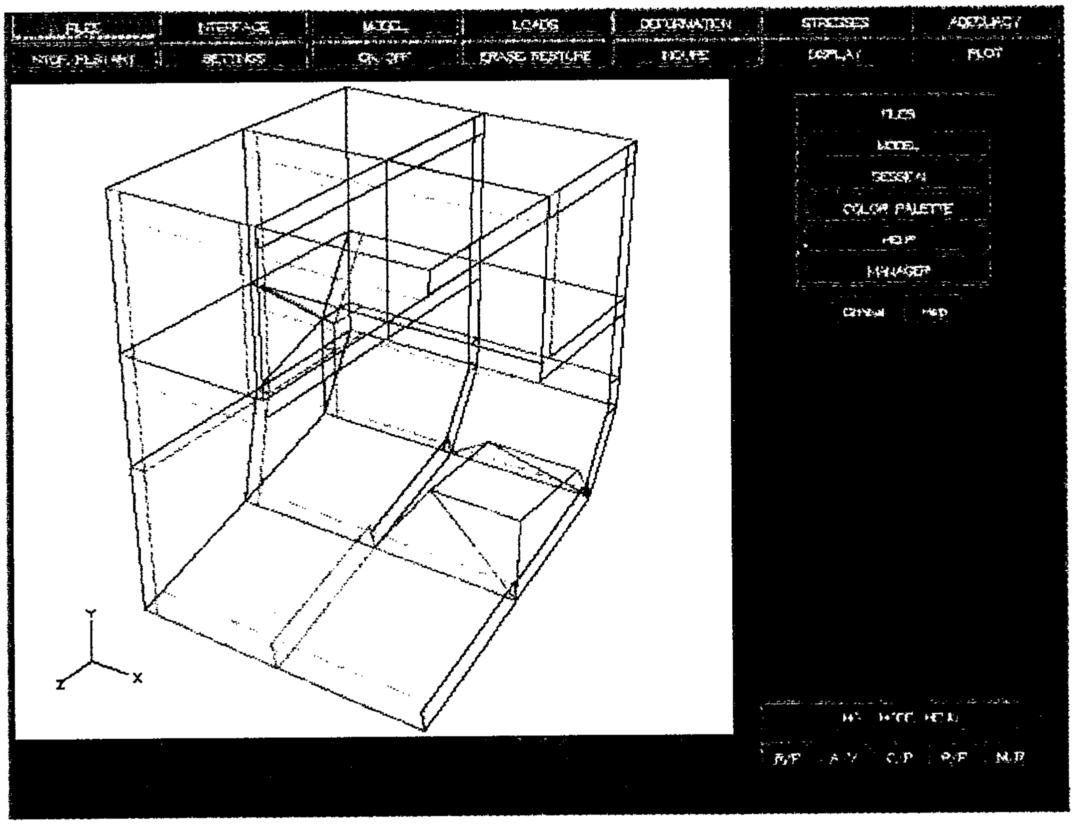

Figure 4: WIRE MESH Plot of the Model EX1

(5) PLOT was chosen and then the button FILL HIDE produced a panel fill with hidden line as shown in figure 5.

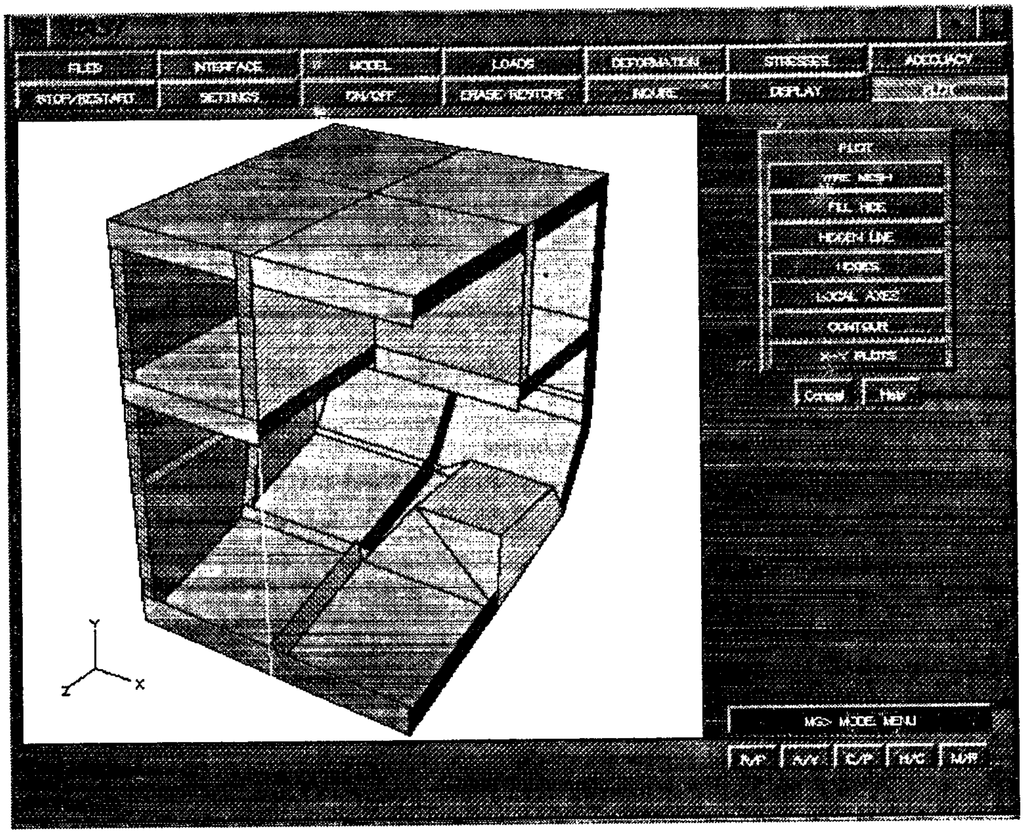

Figure 5: A FILL HIDE Plot of MAESTRO Model

(6) The loads on the model were shown by first clicking on LOADS. The load case was entered on the appearance of the auxiliary menu. The length of the load vectors was determined by the default value 0. A click on OK resulted in the display of the vectors as shown in figure 6.

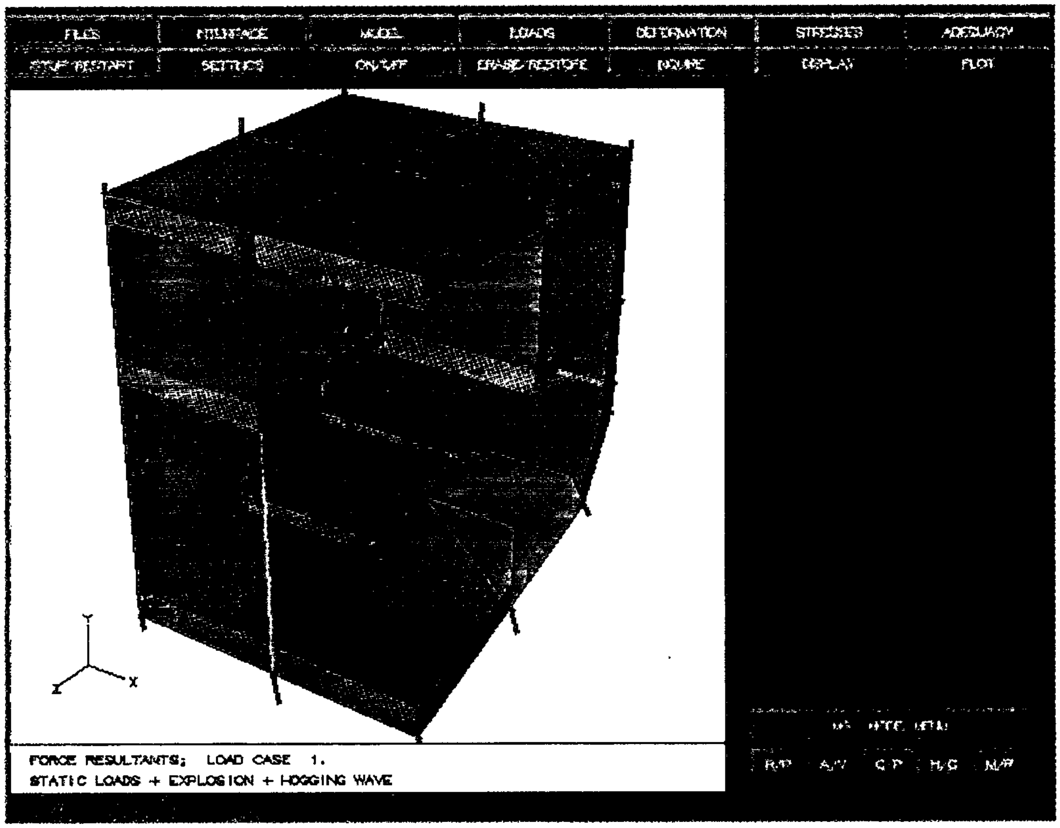

Figure 6: The Loads Applied to the Model

(7) The displacements due to the loading were obtained by first selecting DEFORMATION then DISPLACEMENTS. If the load case number had not been previously entered it could be entered at this stage. A click on PLOT gave the deformed plot of figure 7.

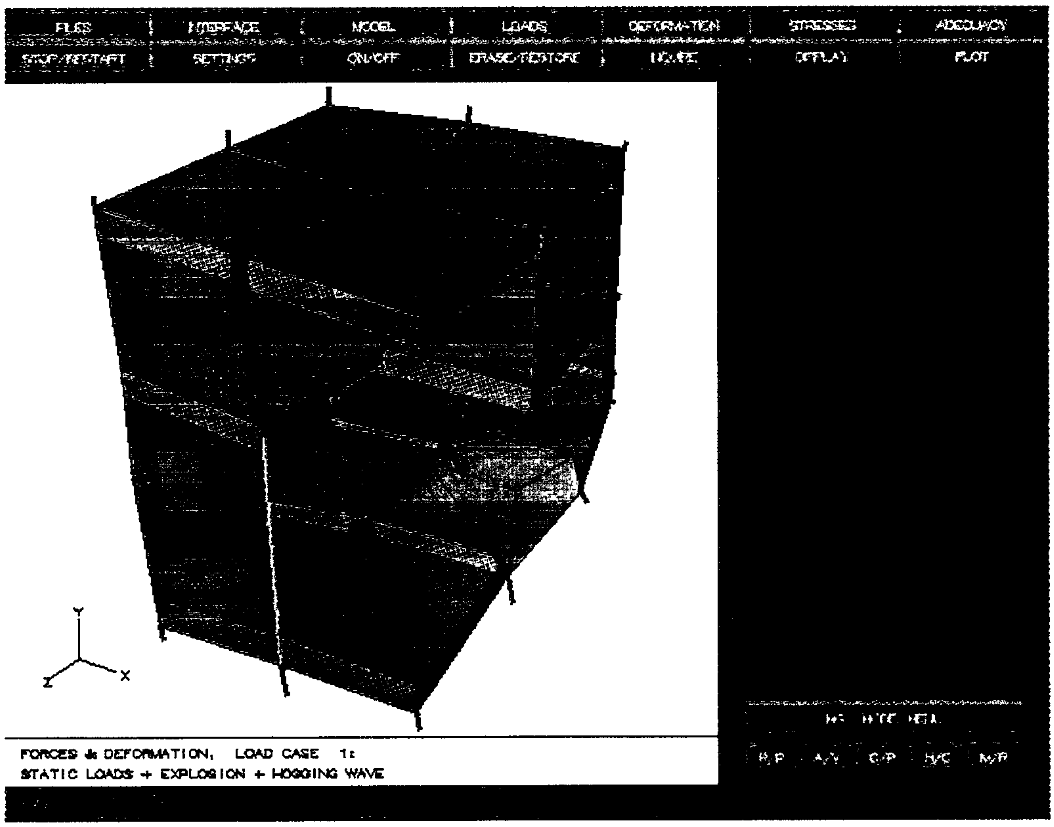

Figure 7: The Distortion of the Model Due to Loading

(8) Displacement contours were obtained by selecting PLOT followed by CONTOUR then DISPLACEMENTS. The contours are shown in figure 8.

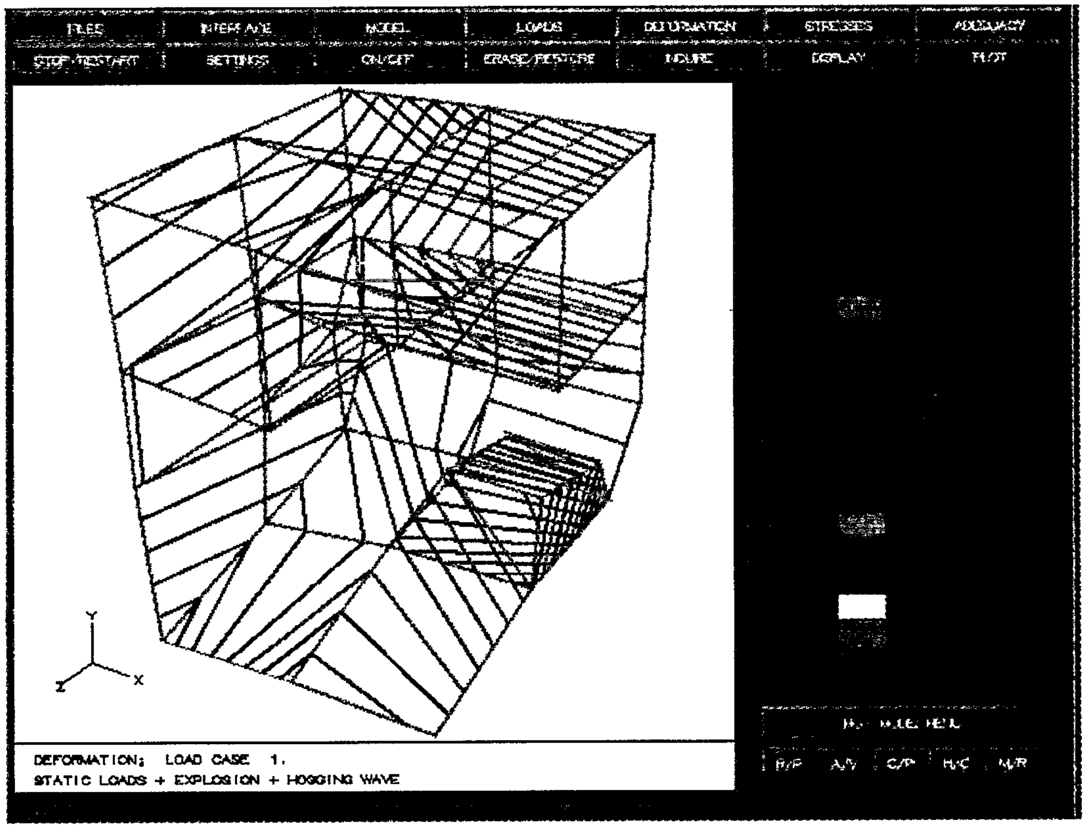

Figure 8: The Displacement Contours Due to Loading

(9) The STRESS button was activated to obtain the stresses. The auxiliary menu STRESSES displayed the load case number. The load case number could be changed at this stage. The MAESTRO element for which the stresses were required was the STRAKE PANEL. The stress, 'Element - Von Mises', was chosen. The maximum stress of 172 Mpa, as shown by the colour coded panels in figure 9, occurred in the bottom panels next to the keel.

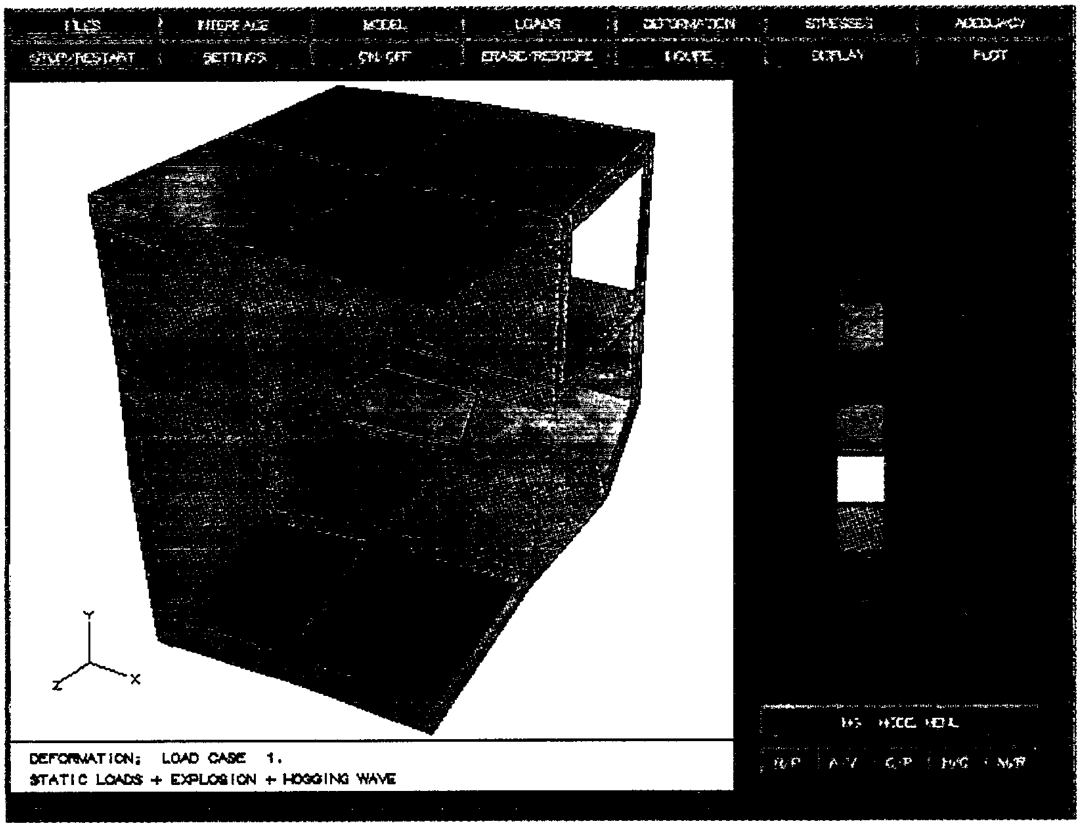

Figure 9: The Von Mises Stresses in the Model Due to Loading

(10) Stress contours were obtained by clicking on PLOT then CONTOUR followed by STRESSES. The stress distribution as shown in figure 10 is better defined with the maximum stress of 167 Mpa occurring near the keel in the two bottom panels and a stress of 156 Mpa further away thereby showing a stress gradient not seen in the colour coded panels.

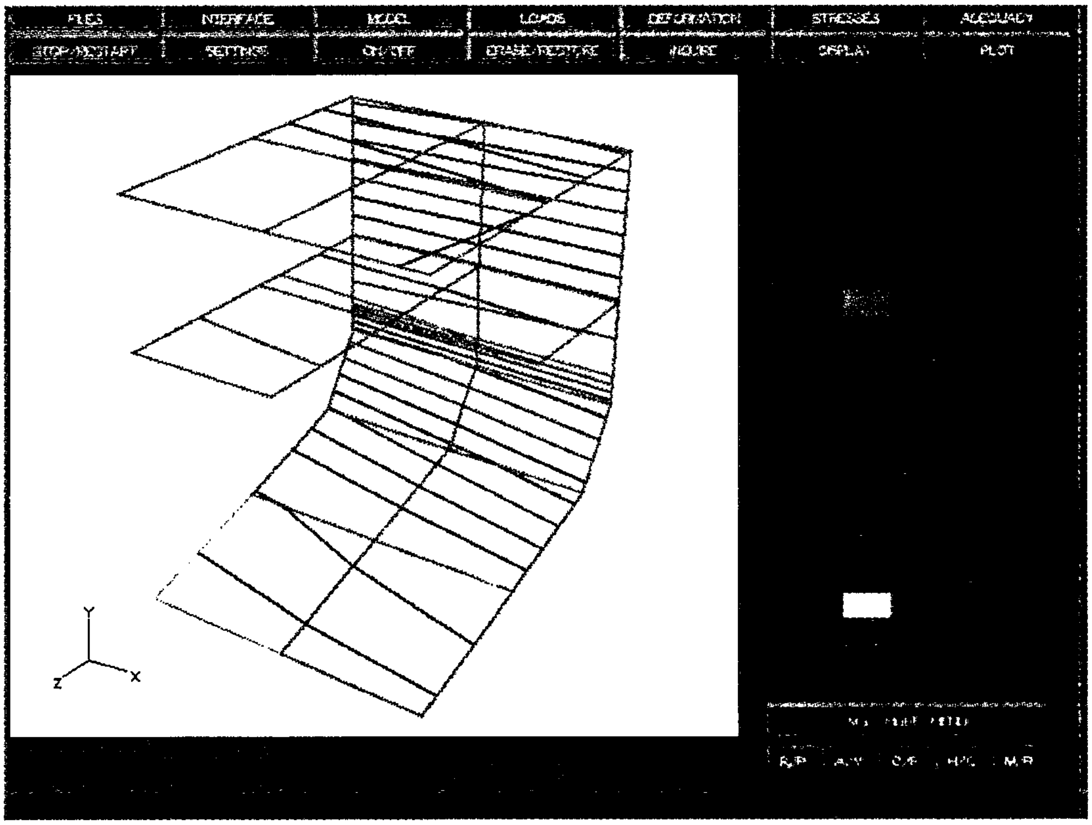

Figure 10: The Von Mises Stresses Contours in the Model Due to Loading

3 Terminal Session of the Bottom-up Analysis

The two bottom panels with the highest stress in the model were chosen for refinement for a superelement or bottom-up analysis using the DSA mode for the detailed stress analysis.

(11) The distortion due to loading was removed from the model by activating the ON/OFF button. The 'Displaced Shape' was set OFF and the 'Stiffener Display' was set ON. The model was then plotted with FILL HIDE displaying the stiffeners as shown in figure 11.

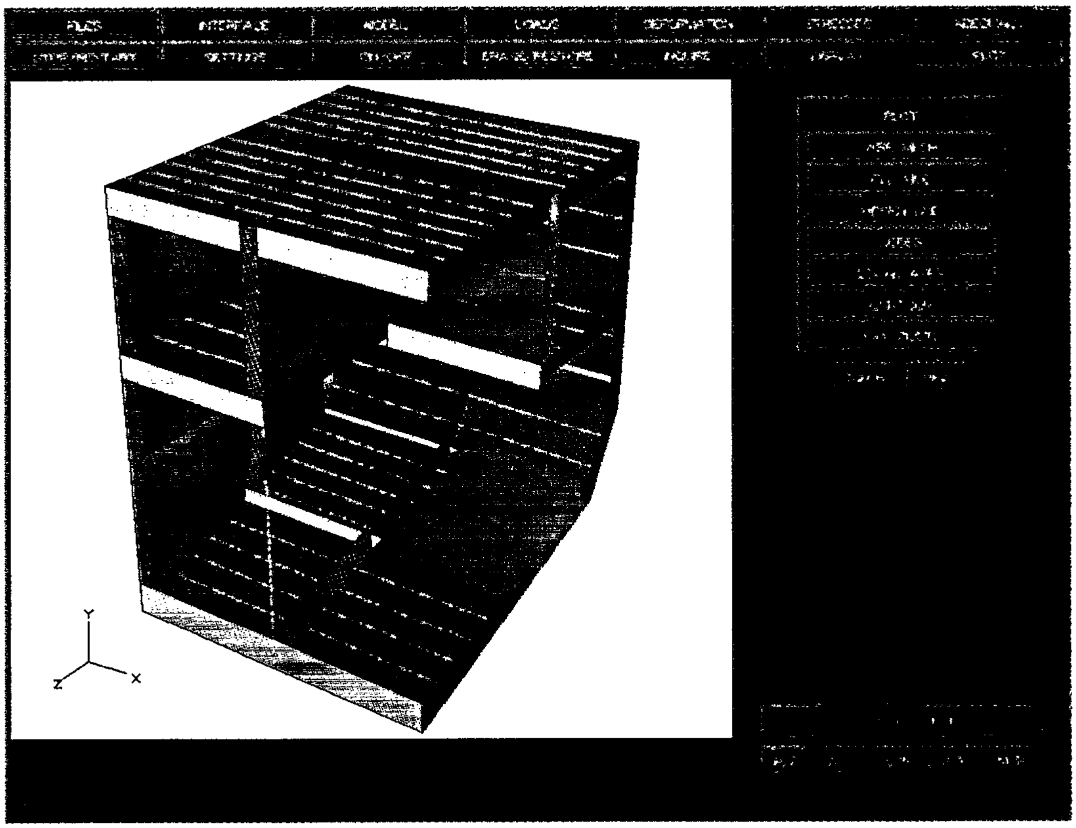

Figure 11: The Model Plotted With the Stiffeners Displayed

(12) The DSA mode was chosen to carry out the detailed or superelement analysis. GEN/MOD was selected followed by REFINE then ELEMENT in the secondary menus. The QUADRILATERAL element was chosen for the refinement. The prompt at the bottom of the screen, indicating a MAESTRO element was to be refined, was activated. The centroid of the bottom panel was then indicated and a mesh was shown which could be accepted or changed by the user. When the OK button was activated the mesh was displayed on the chosen panel. The option to equivalence and compress the new nodes was refused at this time. Three panels were gridded in this way. This was followed by the use of the 'Line' option to grid the eight surrounding MAESTRO beams, then changing the default value for the beam refinement from 10 to 5 to match the plate grid. The detailed portion of the model created by the refinement is shown with the complete model in figure 12.

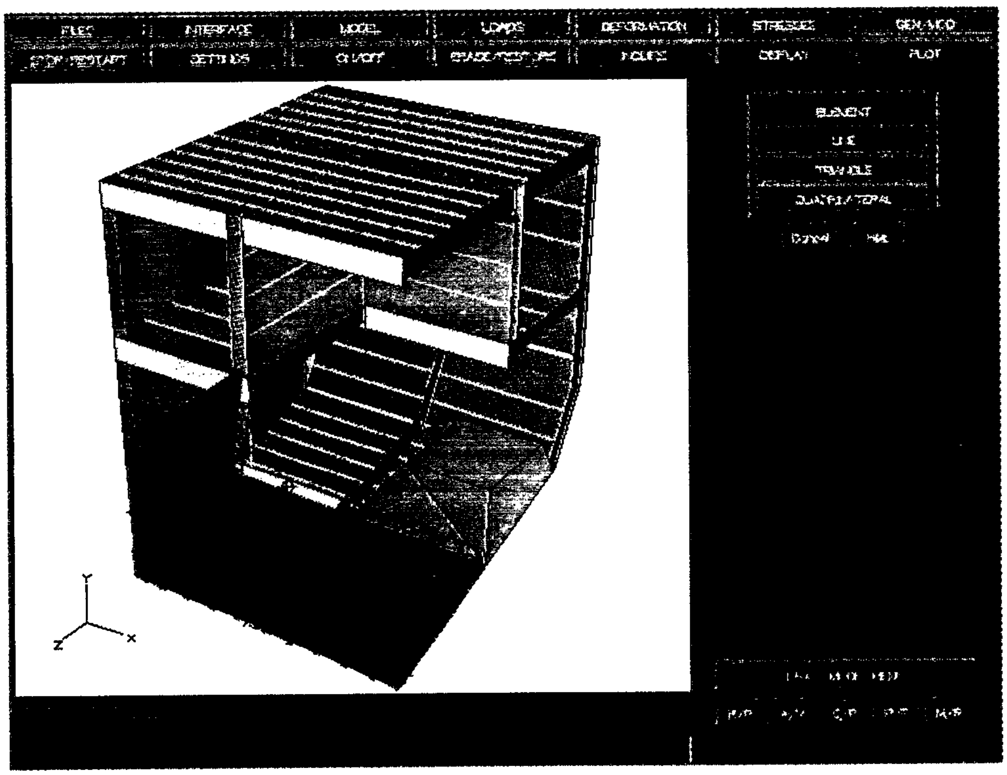

Figure 12: The Refined Panels and Beams Forming the Detailed Model

(13) The next step in the procedure was to equivalence and compress the new nodes. This was accomplished by GEN/MOD followed by NODES in the auxiliary menu. REGULAR was then chosen in the NODE menu. At this stage the nodes were equivalenced and compressed by clicking on EQUIVALENCE then COMPRESS. It was important that this was done before the master nodes and slave nodes were created to avoid a misplacement of these nodes by the equivalencing and compressing process. The resulting model is shown without the clutter of the refinement procedure in figure 13.

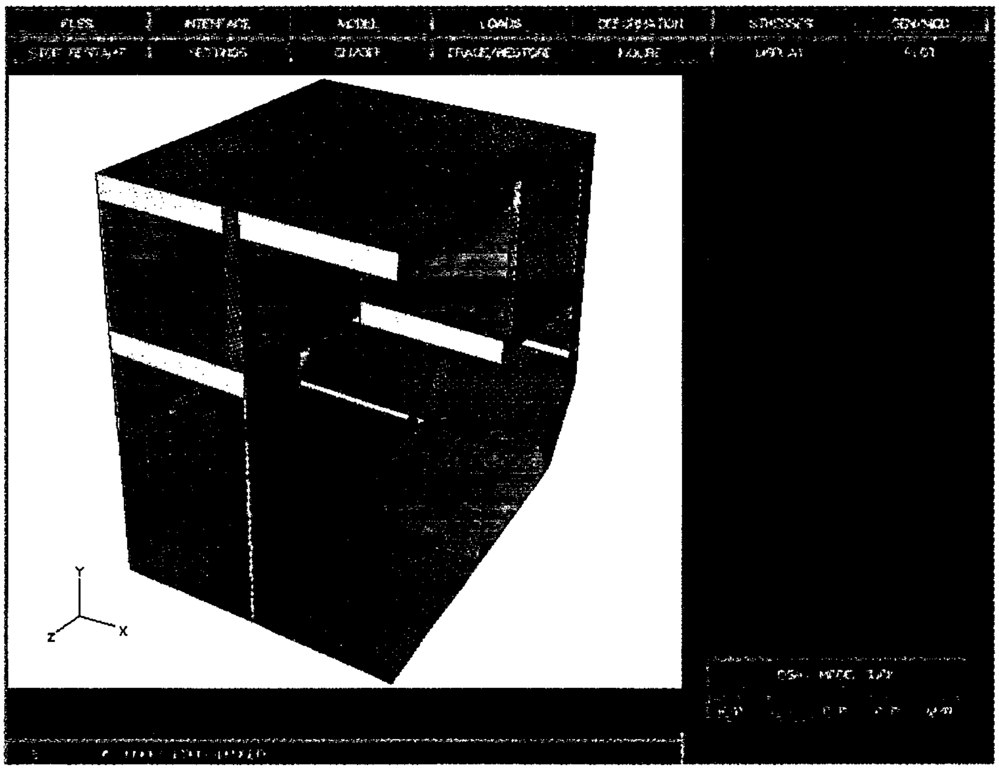

Figure 13: The Model with the Refined Portion after Equivalencing

(14) The beams that were created are shown in better detail, in figure 14, by plotting using the SETTINGS options SHRINK and STRUT WIDTH set to 20 and 5.

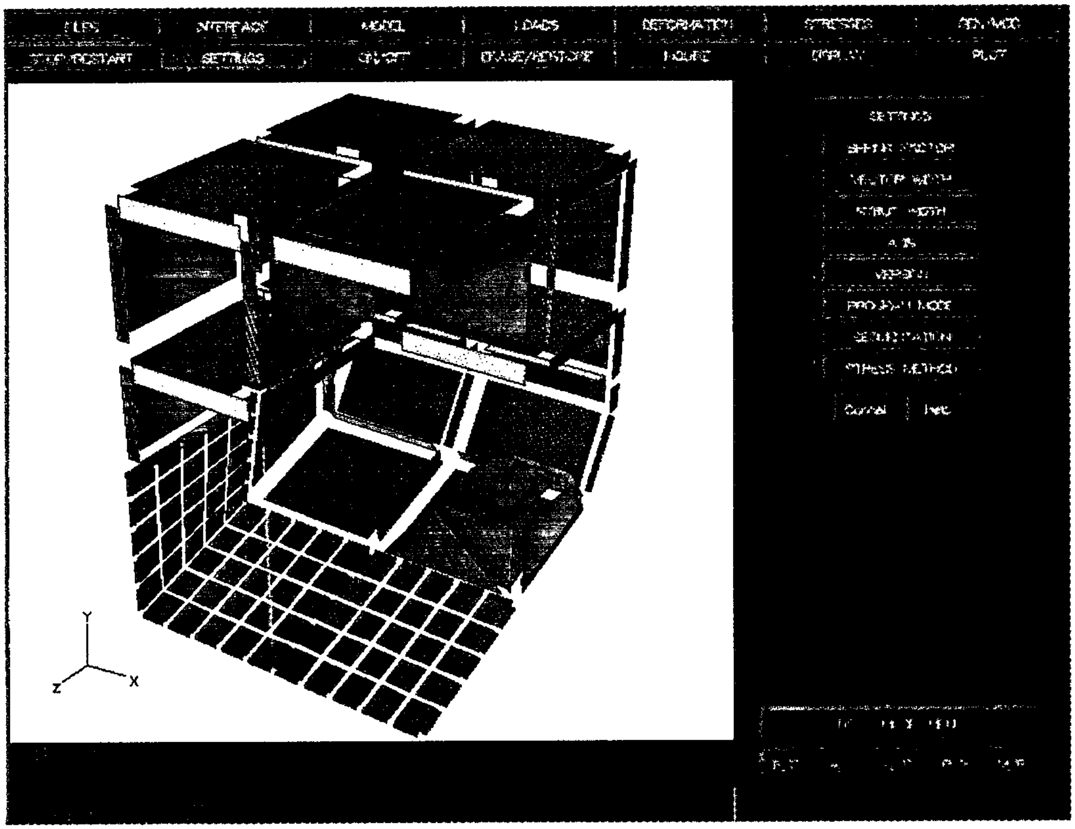

Figure 14: A Plot of the Model with the Refined Beams made Visible

(15) The refined portion is shown in the wire model in figure 15. It was displayed by plotting the MAESTRO model using the WIRE MESH option with SHRINK set to 0 and STRUT WIDTH set to 1 . After this process the GEN/MOD menu was selected. The NODES button was activated producing the NODES menu. The MASTER button was selected to create the master nodes. The default 'Create' was displayed and accepted by clicking on OK. The option to use common MAESTRO nodes was selected with a tolerance of 0.1. The master nodes were displayed and the master node count given. A click returned the NODES menu and SLAVE was selected. The SLAVE NODES menu appeared. The 'Single/Line/Box' was clicked to 'Line' followed by OK. The slave nodes were created by clicking on the dependent master nodes. Once they were created they could be displayed at any time by selecting PLOT and either WIRE MESH or FILL HIDE then NODES. The results of the master and slave creation can be checked by clicking on MODEL then SUMMARY.

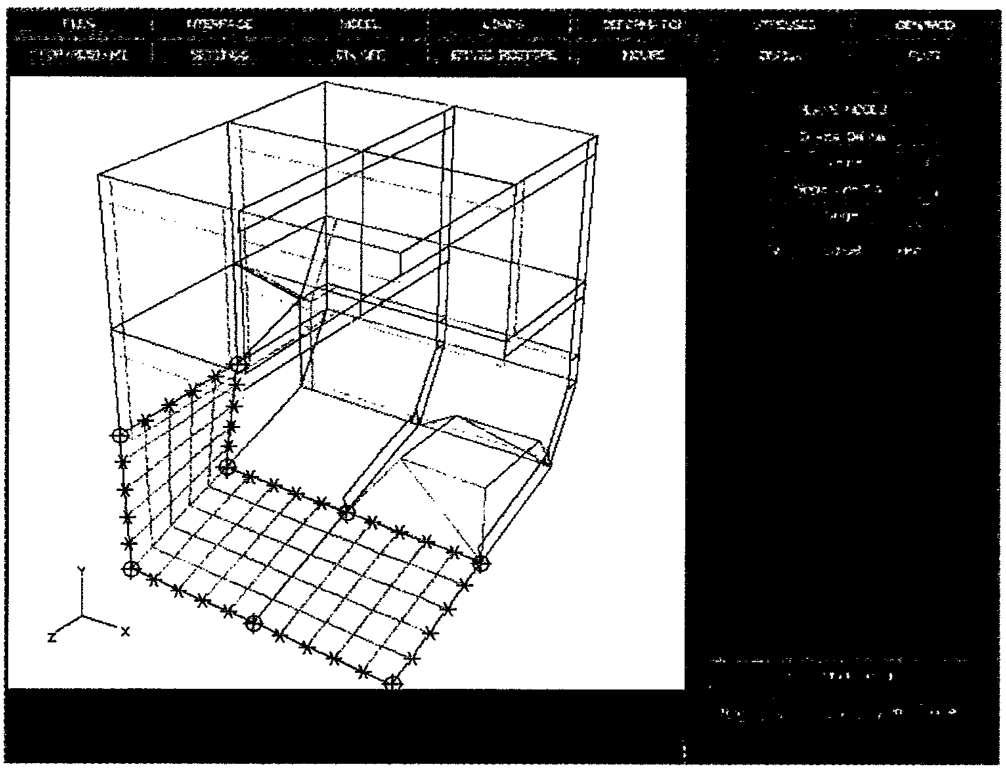

Figure 15: The Refined Portion and the Master and Slave Nodes Shown as Part of the Wire Model Plot

(16) The boundary conditions were checked at this stage by clicking on MODELS then BOUNDARY CONDITIONS to produce figure 16

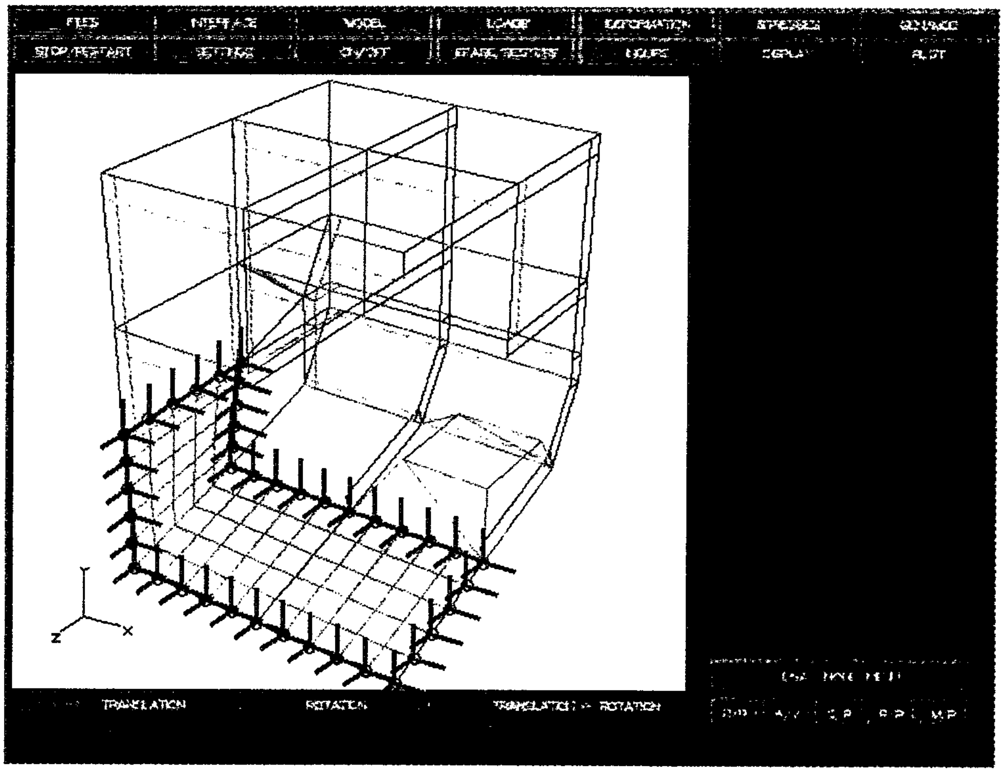

Figure 16: The Boundary Conditions Applied to the Model

(17) The DSA files were then created by choosing INTERFACE on the main menu, MAESTRO/DSA on the secondary menu, and SUPERELEMENT ANALYSIS on the next menu. The option to CREATE DSA FILES was chosen on the next menu. In this case the default DSA prefix 'dsase' was used instead of a user defined prefix and files dsase.gom, dsase.sed, and dsase.smd were created. The refined portion was plotted separately by choosing FILE then MODEL followed by OK. The option to append to the existing model appearing at the bottom left of the window was refused, and the options to read the superelement and boundary condition files were accepted. PLOT then FILL HIDE displayed the refined portion without the remainder of the model as shown in figure 17.

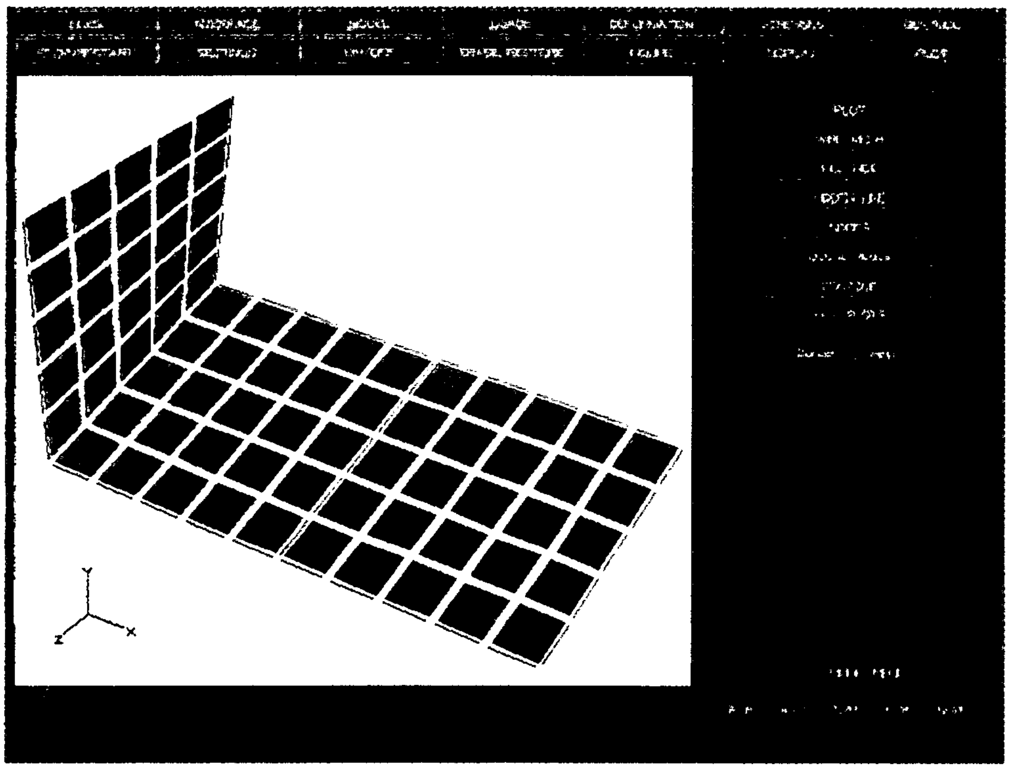

Figure 17: The Refined Portion Plotted Separately

(18) At this stage the program was stopped by STOP/RESTART. The superelement analysis was then carried out. In this process the refined portion of the model was automatically converted to a superelement and its master nodes connected to the equivalent MAESTRO nodes. The job name EX1 and the default file name 'dsase' were used in this case. On the completion of the run the deformations were plotted. The MAESTRO model was plotted first using the MG option then FILE then MODEL then FILL HIDE. This ensured the model was available for plotting. The DSA mode was activated and INTERFACE, then MAESTRO/DSA followed by SUPERELEMENT were selected. When the SUPERELEMENT ANALYSIS menu appeared, 'Read DSA File' was chosen and the job name EX1 was OK'd. DEFORMATION was clicked on and DISPLACEMENTS selected. OK was clicked on to approve the DSA prefix name 'dsase'. The load case was confirmed and the PLOT of the deformations was displayed with FILL HIDE as shown in figure 18.

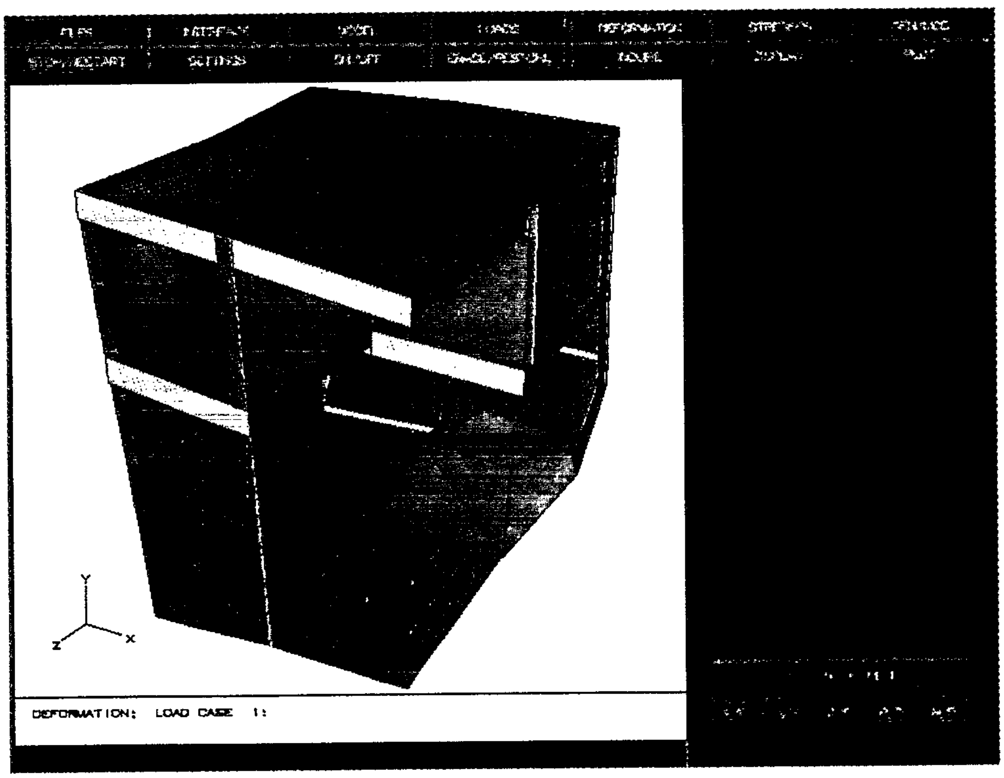

Figure 18: Deformations of the Superelement as Part of the Full Model

(19) The deformations of the superelement displayed as part of the wire model are shown in figure 19. They were obtained by selecting PLOT and then WIRE MODEL.

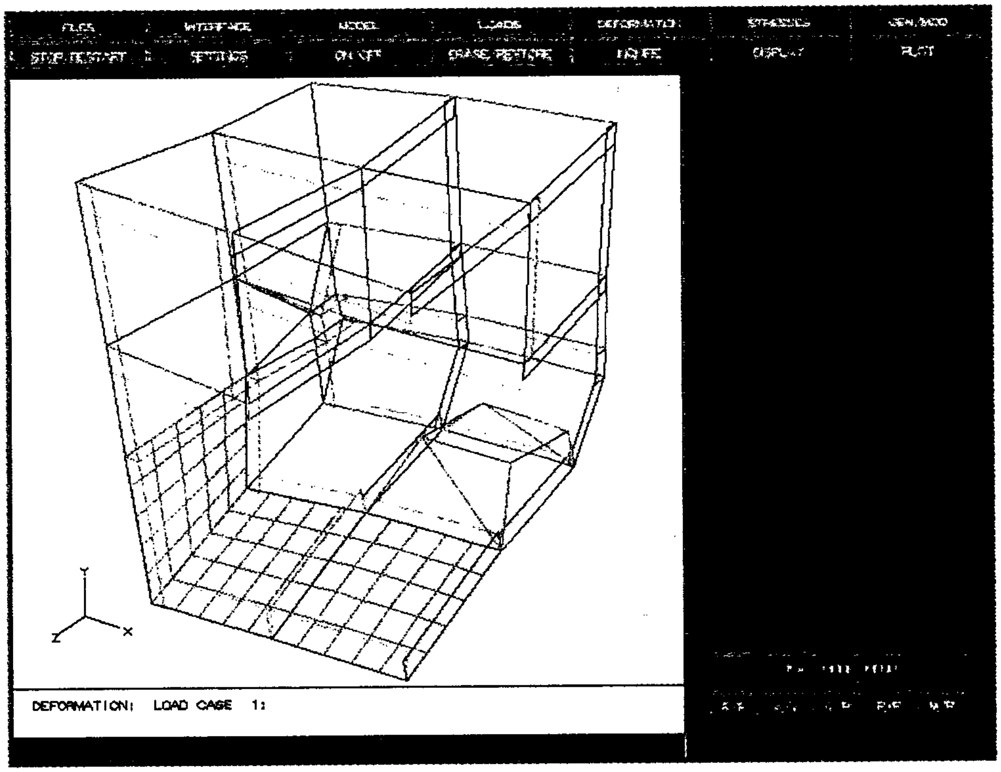

Figure 19: Deformations of the Superelement as Part of the Wire Model

(20) The distortion of the model in the form of displacement contours was obtained by selecting CONTOURS under the PLOT options and is shown in figure 20.

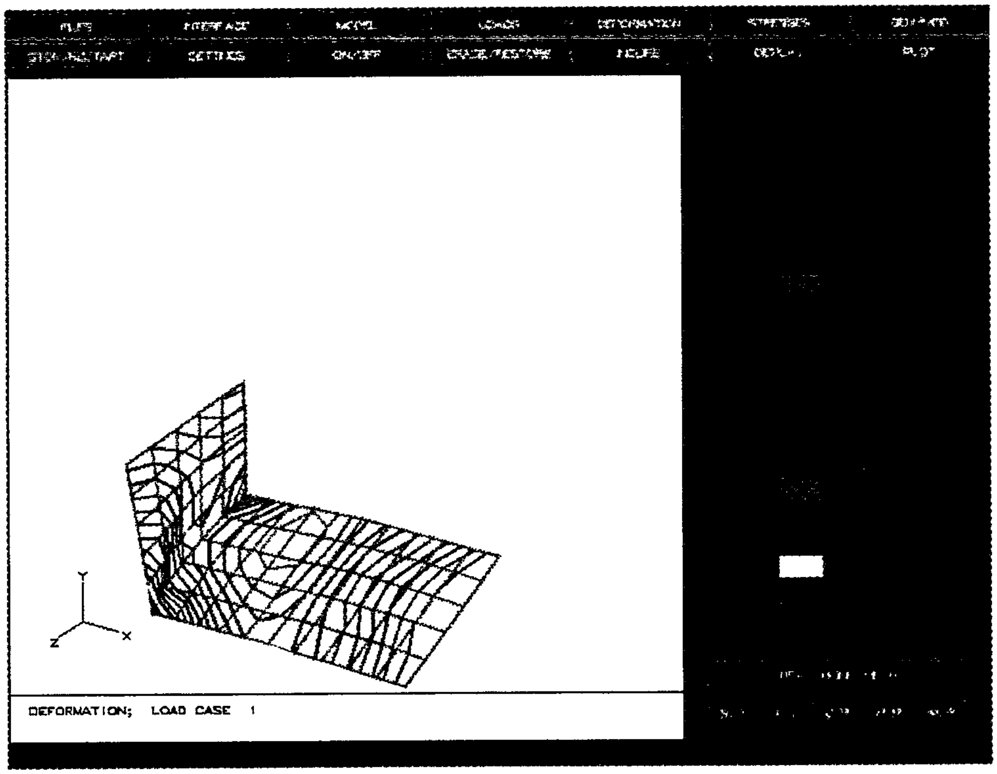

Figure 20: Deformatioms of the Superelement in the form of Displacement Contours

(21) The stresses were displayed by selecting STRESSES, then OK'd for the load case followed by QUADRILATERAL SHELL and then Von Mises stress. PLOT then FILL HIDE resulted in the colour stress plot shown in figure 21. The maximum stress was 369 MPa.

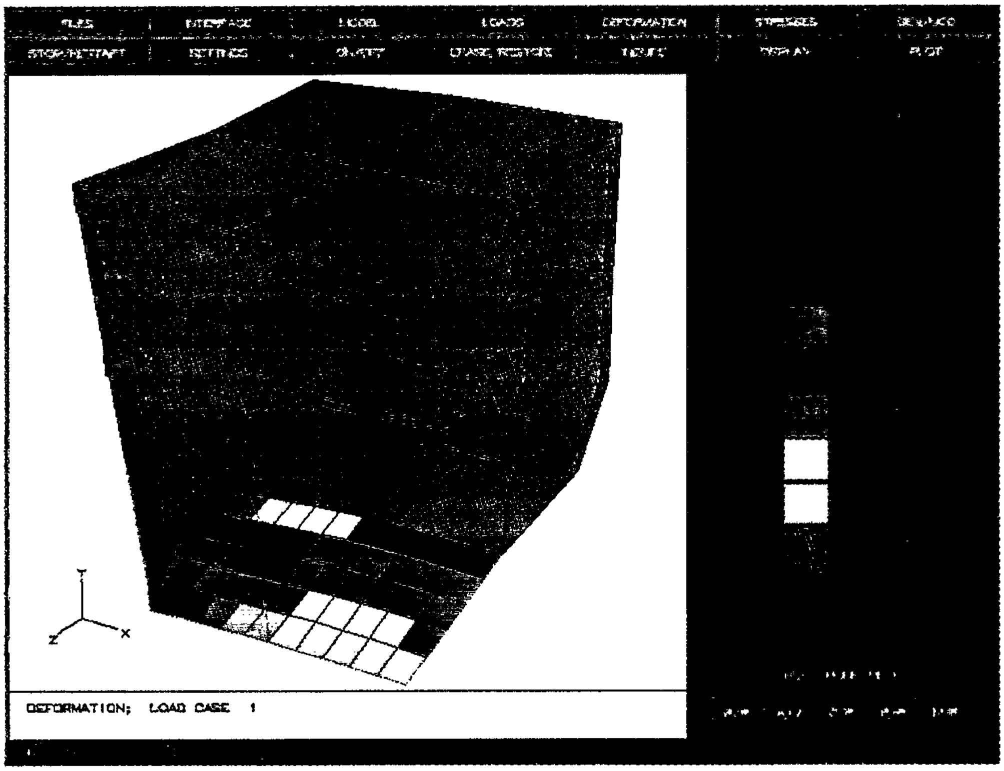

Figure 21: Colour Plot of Stresses in the Superelement

(22) A contour plot of the Von mises stresses in the element shown in figure 22 was obtained by selecting CONTOUR in the PLOT menu. The maximum stress in the contour plot is 347 MPa.

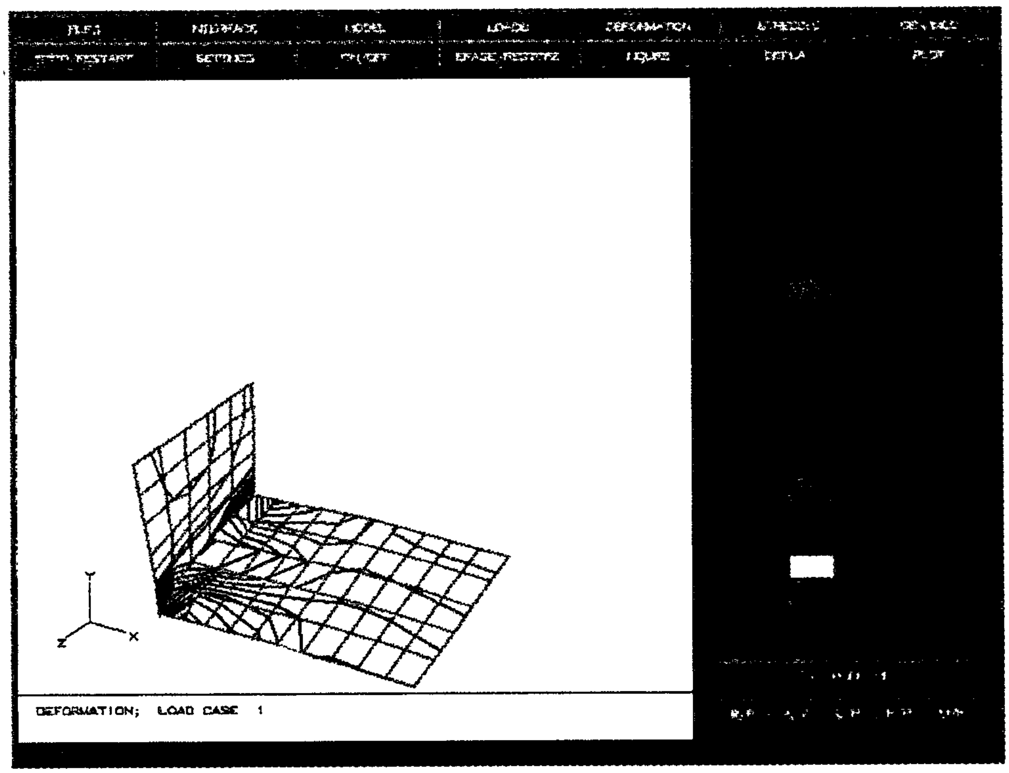

Figure 22: Contour Plot of Stresses in the Superelement

(23) A colour plot of the stresses in the superelement element is shown plotted separately in figure 23. It was obtained by choosing the MG mode then ERASE/RESTORE followed by SUBSTRUCTURE/MODULE. Then OK was given to the 'Table' and the substructure was then turned off in the table list, thereby removing the MAESTRO model from further plotting. The DSA mode was chosen, STRESSES was selected, and the procedure described in (21) was followed.

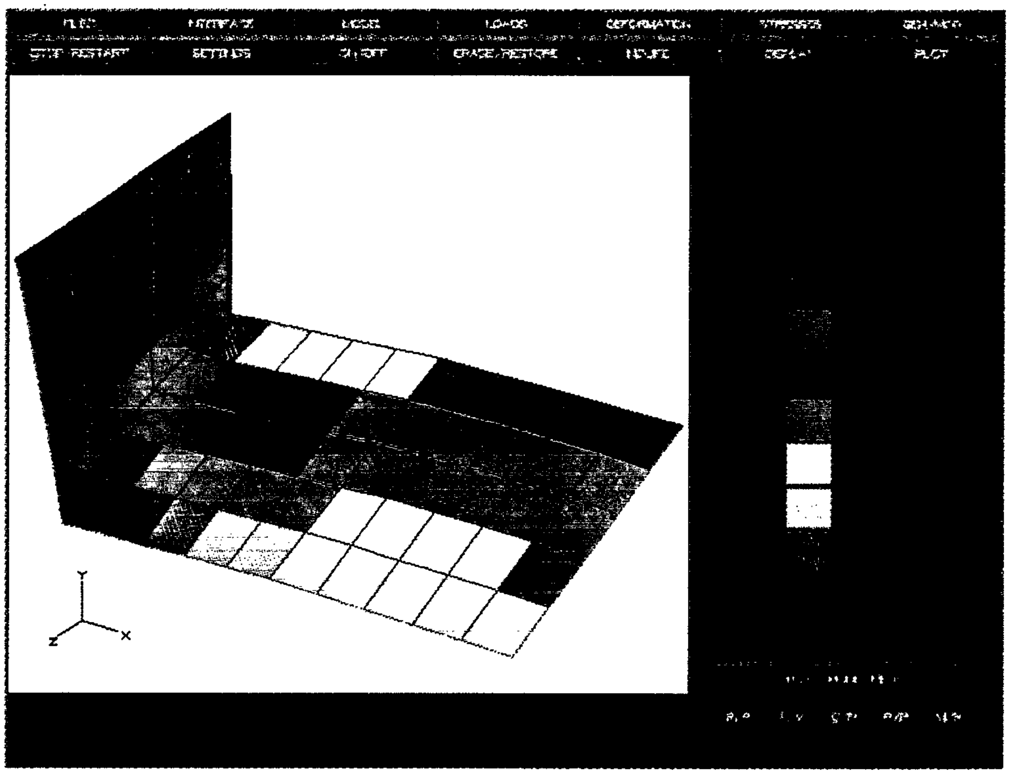

Figure 23: A Colour Map of Stresses in the Superelement Plotted Separately

(24) The stress contours in the superelement were plotted separately, as shown in figure 24, by choosing CONTOUR in the PLOT menu.

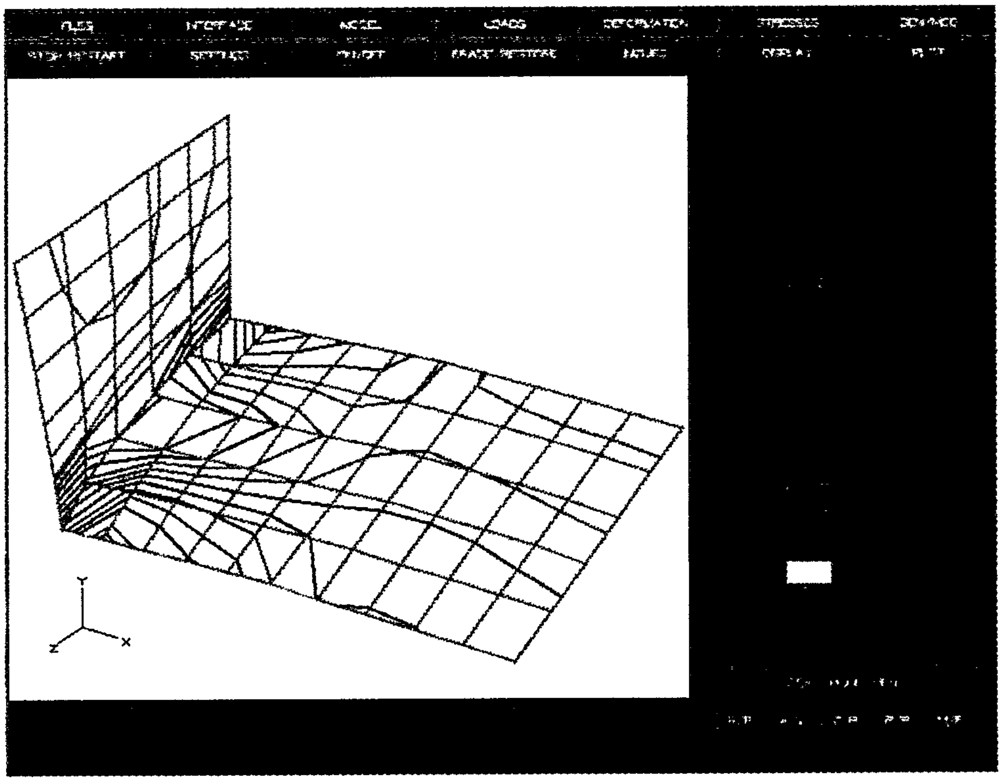

Figure 24: A Contour Plot of Stresses in the Superelement Plotted Separately

(25) The beam stresses were plotted by setting SHRINK to 20 and STRUT WIDTH to 5 then selecting STRESSES. The auxiliary menu ELEMENT TYPES was displayed from which BEAM was chosen and the crossection stress at the four corners of the beams were examined. The highest stress of -527 Mpa was found at point 4 end 2 as shown in figure 25.

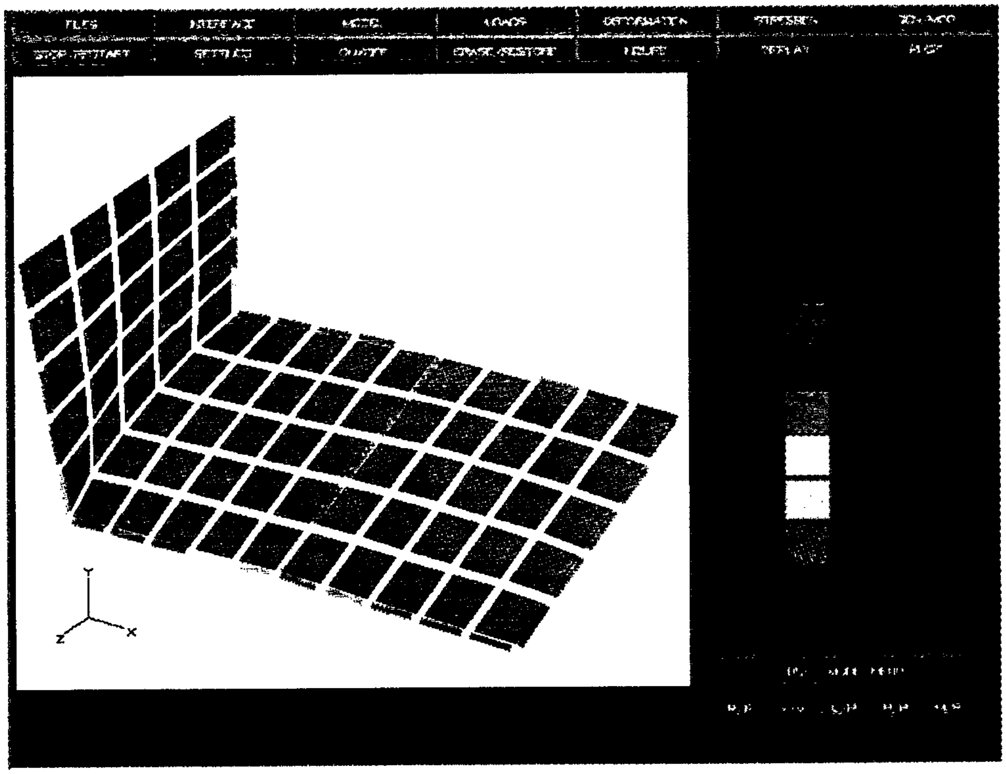

Figure 25: A Plot of Beam Stresses at Point 4 End 2 in the Beam Crossection

4 Terminal Session of the Top-Down Analysis

The top-down analysis used the same detailed model as was used in the bottom-up method but it was not made a superelement where its stiffness matrix was made part of the MAESTRO equilibrium equations. Instead the displacements at the identified master nodes, resulting from the MAESTRO analysis, were applied to the master nodes of the detail model as prescribed displacements, and a stand-alone finite element analysis was carried out to obtain the stresses.

(26) In the top-down analysis INTERFACE was activated followed by MAESTRO/DSA, TOP DOWN ANALYSIS, and 'Create Dsa File'. The program was then stopped by STOP/RESTART. Then MAESTRO was run and the detailed stress analysis was chosen. On its completion MG/DSA was run again and FILE was activated. Then MODEL was chosen and the job file name EX1 was entered. The DSA mode was switched on and INTERFACE was selected followed by MAESTRO/DSA and TOP DOWN ANALYSIS. Then 'Read Dsa File' was chosen. PLOT and FILL HIDE displayed the view shown in figure 26.

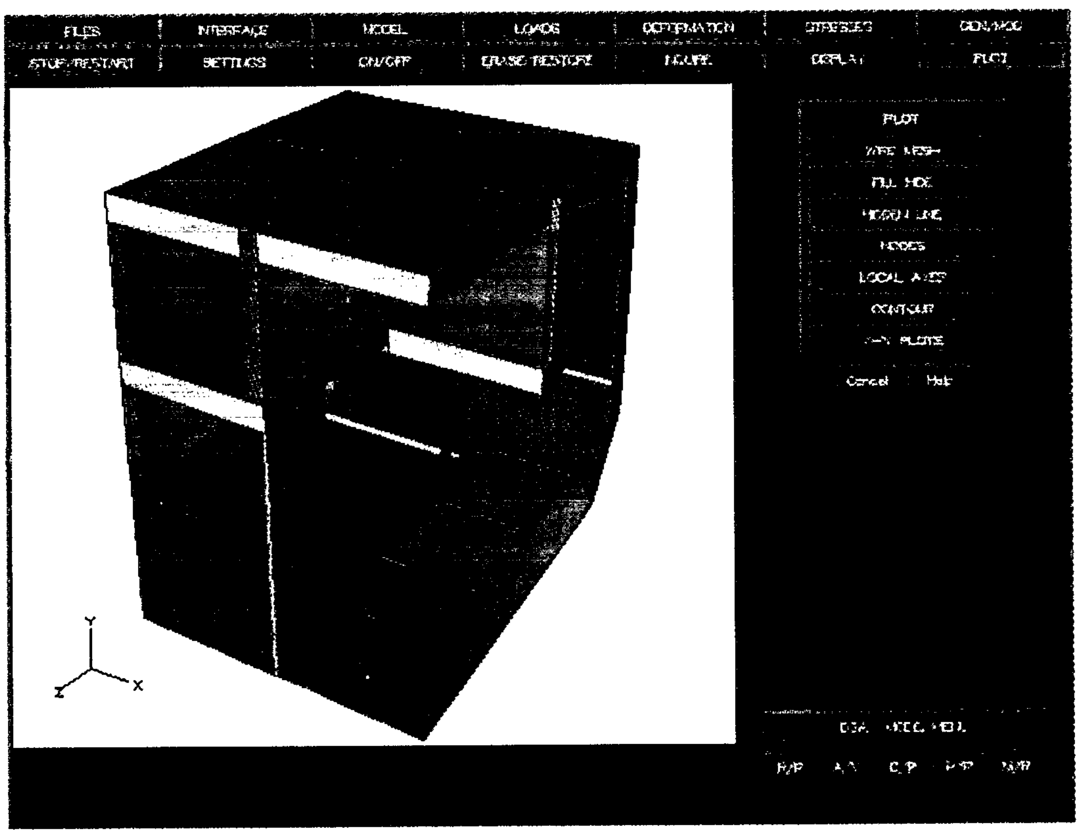

Figure 26: A Plot of the Top-Down Model

(27) The deformations obtained from the top-down analysis were displayed by clicking on DEFORMATION then OK to file name 'topdwn' given as the DSA prefix. Load case 1 was OK'd followed by FILL HIDE to produce figure 27.

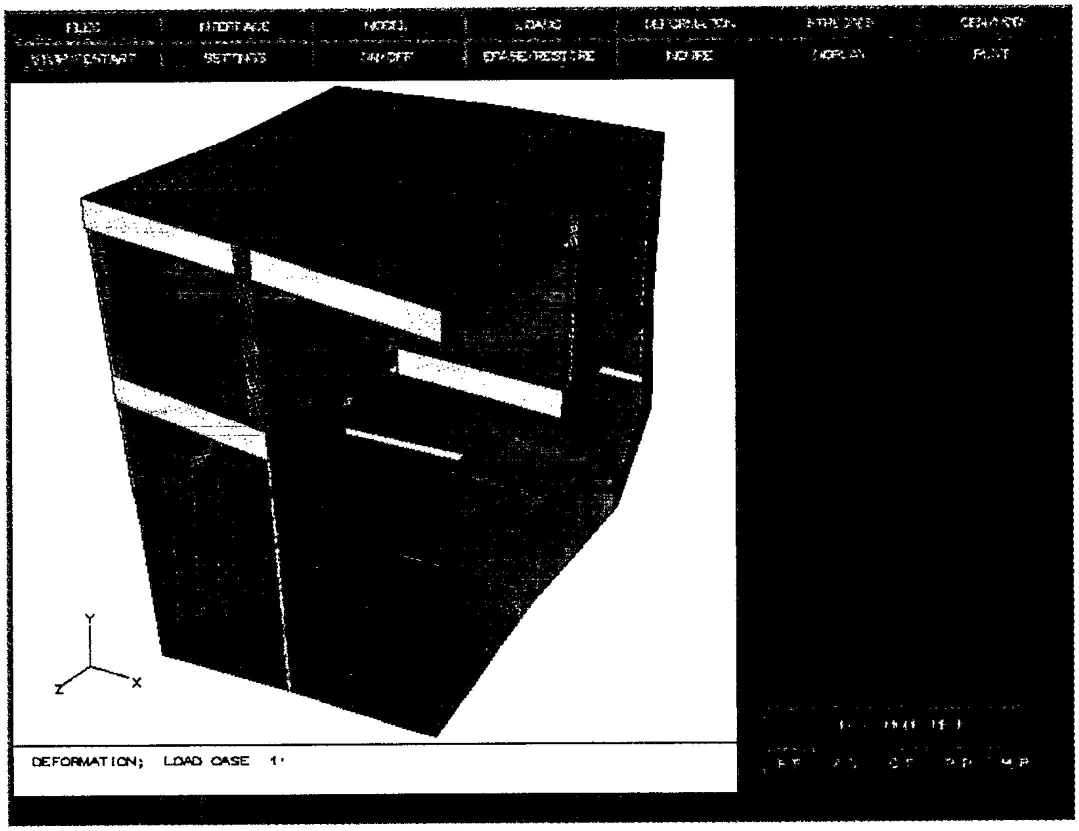

Figure 27: A Plot of the Deformed Model Resulting from the Top-down Analysis

(28) The displacement contours where displayed as shown in figure 28 by clicking on CONTOURS in the PLOT menu.

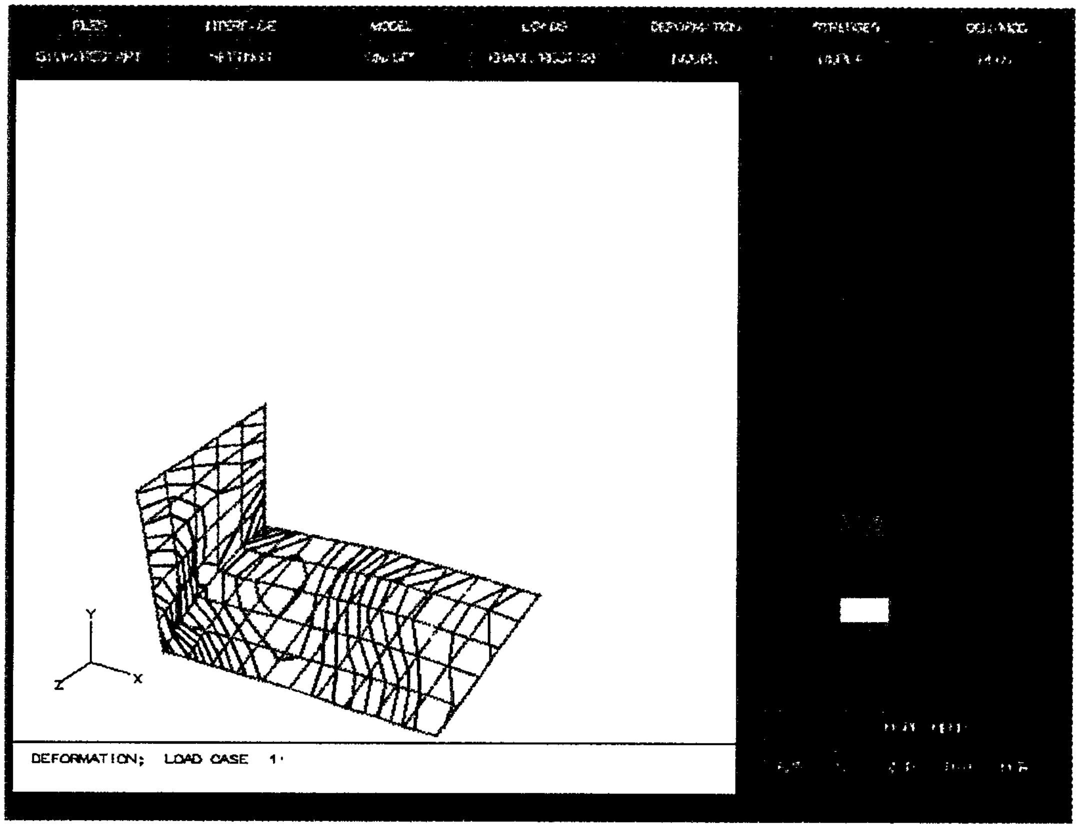

Figure 28: The Displacement Contours from the Top-down Analysis

(29) The stresses from the top-down analysis were plotted by clicking ON/OF to switch off the 'Displaced Shapes'. This was followed by STRESSES then OK to 'Load Case 1'. QUADRILATERAL SHELL was chosen along with 'Von Mises Stress' to produce the coloured stress map of figure 29.

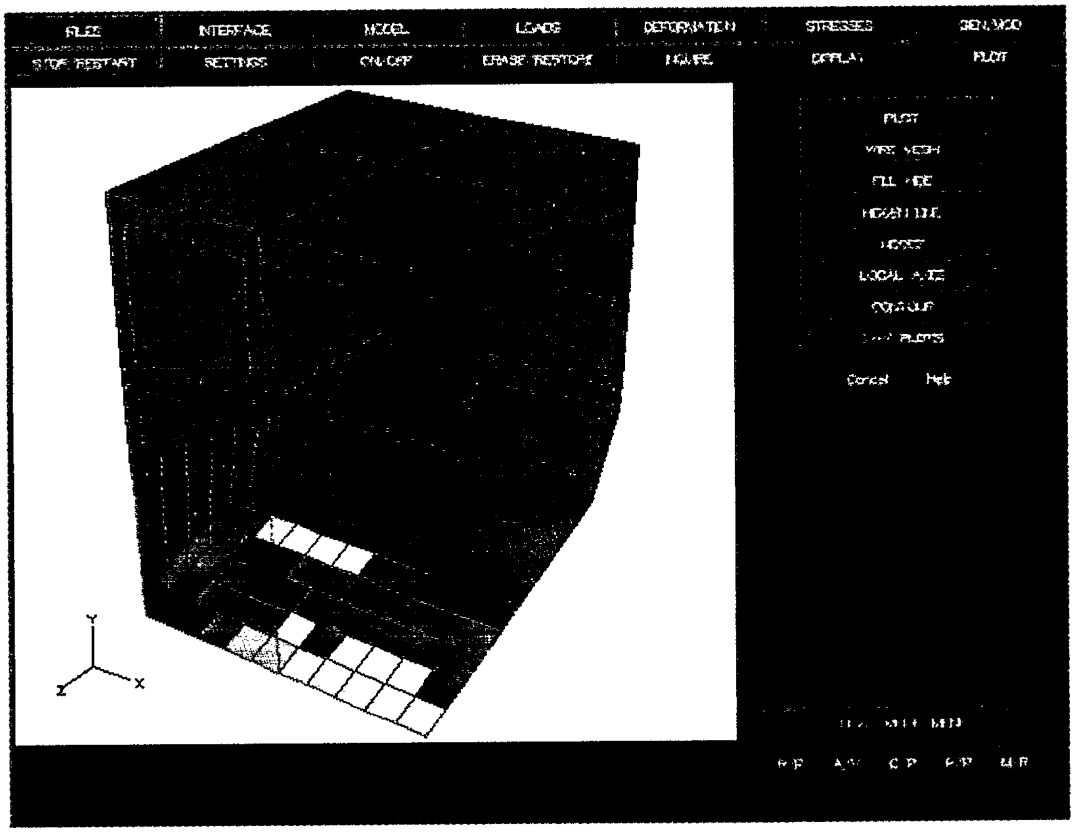

Figure 29: The Colour Stress Map from the Top-down Analysis

(30) The stress contours were obtained by selecting CONTOUR from the PLOT options followed by 'stress' from the contour options to produce figure 30. The maximum stress was 348 MPa.

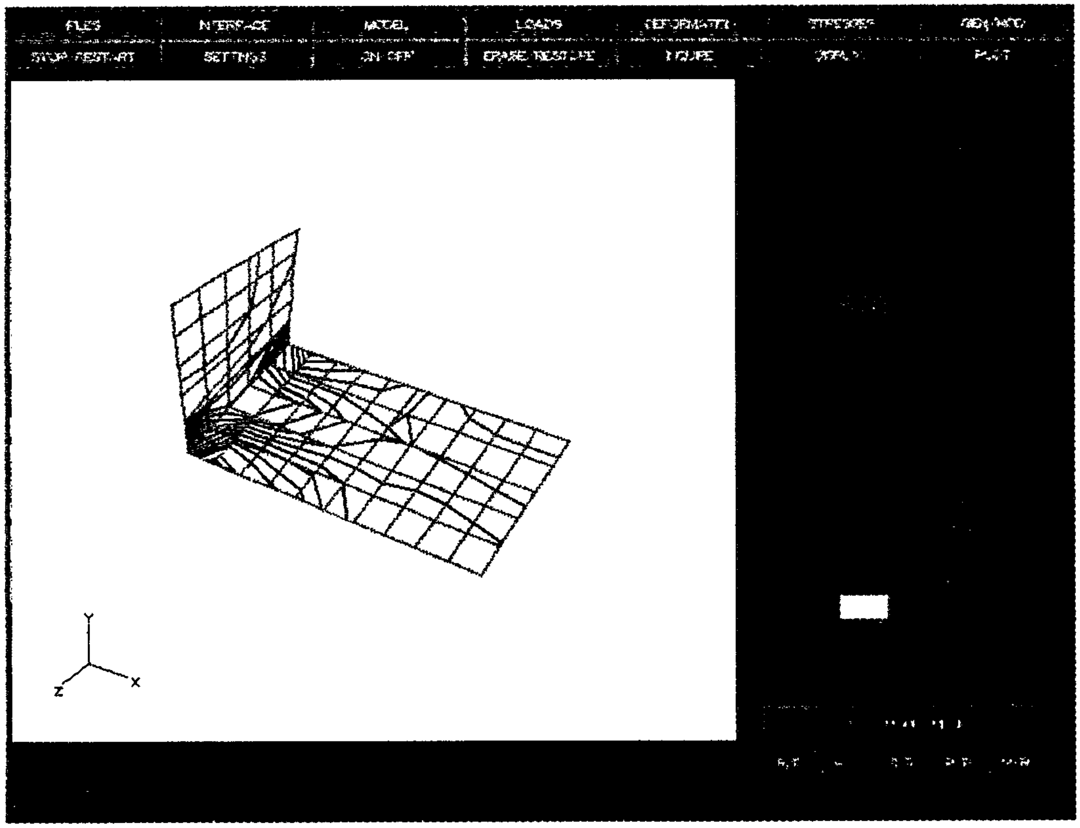

Figure 30: The Stress Contours from the Top-down Analysis

(31) The beam stresses from the top-down analysis were obtained by setting the MG MODE and clicking on SETTINGS. Then SHRINK was set to 20 and STRUT width to 5. ERASE/RESTORE was clicked on, and SUBSTRUCTURE MODULE was selected and OK was given to 'Table'. In the table SUBSTRUCTURE 1 was removed to allow the top-down model to be plotted alone. The DSA MODE was set followed by STRESSES, and OK was given to the load case. Then BEAM was selected from the element types. The 'Combined Point 4, End 2' was selected for the beam cross section stress location and PLOT then FILL HIDE produced the beam stress plot shown in figure 31. The maximum stress in the beams was -532 MPa at point 4 of end 2 of the purple coloured beams.

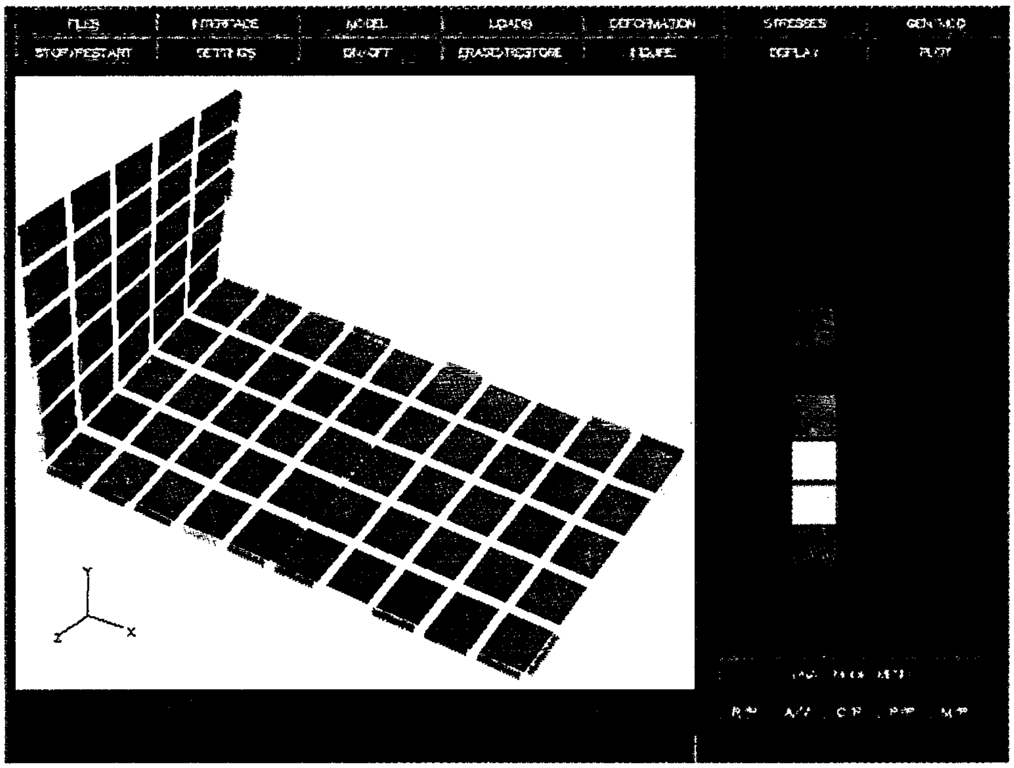

Figure 31: The Stresses at Point 4 End 2 in the Beams after the Top-down Analysis

5 Conclusions

Both a bottom-up and a top-down analysis were carried out on a portion of a ship structure modelled with the structural analysis program MAESTRO. The panels identified as the most highly stressed were refined, using the program MAESTRO/DSA, to obtain a greater appreciation of the detailed stresses. Both the bottom-up and the top-down methods gave the reasonably close results of 347 and 348 MPa for the plating, as shown in the stress contours of figures 22 and 30, compared to the MAESTRO results of 167 MPa in figure 13. The results for the beams at end 2 and stress point 4 of the cross-section gave -527 Mpa for the bottom-up analysis and -532 MPa for the top-down analysis as shown in figures 25 and 31. It is concluded that there is little difference between the top-down and bottom-up methods, but that both provide a much better definition of the local stress condition than MAESTRO alone.

References

[1] "MAESTRO-Method for Analysis Evaluation and Structural Optimization, User's Manual-Version 6.0," distributed by Ross McNatt Naval Architects, Annapolis, MD., July 1992.

[2] "MAESTRO Graphics," distributed by Proteus Engineering Stevensville, MD.

[3] "Vibration And Strength Analysis Program (VAST): User's Manual Version 6.0," Martec Ltd., Halifax Nova Scotia, September, 1990.

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (the name and address of the organization preparing the document Organizations for whom the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8.)
Defence Research Establishment Atlantic
P.O. Box 1012
Dartmouth, N.S. B2Y 3Z7

2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable).
Unclassified

3. TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title).
An Example of Obtaining Detailed Stresses of a Portion of a MAESTRO Model of a Ship Structure by Bottom-up and Top-Down Analysis Using MAESTRO MG/DSA

4. AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.)
D.R. Smith

5. DATE OF PUBLICATION (month and year of publication of document)
December 1995

6a. NO OF PAGES (total containing information Include Annexes, Appendices, etc).
34

6b. NO. OF REFS (total cited in document)
3

7. DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered).
Contractor Report

8. SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include the address).
DREA

9a. PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant).
Project 1.g.3

9b. CONTRACT NO. (if appropriate, the applicable number under which the document was written).
W7707-5-3292/01-HAL

10a ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document).
DREA CR 95/500

10b OTHER DOCUMENT NOS. (Any other numbers which may be assigned this document either by the originator or by the sponsor).

11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

(x) Unlimited distribution
( ) Distribution limited to defence departments and defence contractors; further distribution only as approved
( ) Distribution limited to defence departments and Canadian defence contractors; further distribution only as approved
( ) Distribution limited to government departments and agencies; further distribution only as approved
( ) Distribution limited to defence departments; further distribution only as approved
( ) Other (please specify):

12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in 11) is possible, a wider announcement audience may be selected).
Full unlimited

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

13. ABSTRACT (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual).

This report describes the method and results of a detailed analysis of a portion of a ship structure modelled by MAESTRO using bottom-up and top-down analyses. For the bottom-up method, a MAESTRO analysis of the structure was carried out to identify high stress regions. These regions were then modeled in detail using the program MAESTRO MG/DSA. The detail was created as a MAESTRO superelement. The superelement was automatically statically condensed and inserted into the original MAESTRO model equilibrium equations replacing the identified MAESTRO elements. A MAESTRO analysis of the modified model using the same loading and boundary conditions as the original model, was run and detailed displacements and stresses were obtained using the superelement feature of the finite element program VAST. For the top-down analysis, the detail model created for the superelement was used as a stand-alone model. The displacements from the initial MAESTRO analysis were applied to the stand-alone model as boundary conditions and a finite element analysis using VAST was carried out. The results of the three analyses were compared. They showed that a much better definition of the local stress condition was obtained with the detail model than by MAESTRO alone, with little difference in the results between the bottom-up and the top-down methods.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus. e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title).

MAESTRO
VAST
finite element
ship structure
bottom-up
top-down
superelement

UNCLASSIFIED
SECURITY CLASSIFICATION OF FC?M

498675